U.S. Army

Tank-automotive and Armaments Command
TACOM

3 Dimensional Technical Data Package Configuration Management & Modeling Interim Operating Procedure

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Revisions Underlined and Blue

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Chapter 1:
Document Overview

1.1 PURPOSE: This standard describes the requirements and procedures for the Three Dimensional Technical Data Package (3D/TDP) solid model repository established and maintained by TACOM. It is intended that models of TACOM-managed components will be stored in a central location accessible by all TACOM customers, and will be maintained and configuration managed in accordance with standard Army configuration management practices. The 3D/TDP system is based on Pro/ENGINEER Mechanical Design Automation software. The 3D/TDP system uses Pro/INTRALINK product data management software for data management and storage. Pro/ENGINEER and Pro/INTRALINK are both products of PTC (formerly Parametric Technology Corporation). Pro/ENGINEER assigned file extension are .prt for parts, .asm for assemblies and .drw for drawings. Documents which are not solid model related but which normally comprise a TDP such as packaging sheets, quality assurance provisions, electrical schematics, etc. can be stored in the 3D/TDP system.

1.2 RELEASE LEVELS: The database is structured with seven release levels as shown in fig 1. The R&D WIP is the Research and Development Work in Progress Release Level and consists of parts, assemblies and drawings currently under construction, or models being used for conceptualization, prototypes, etc. for TACOM R&D programs. The Pre-Prototype Release Level contains models that the designer has completed initial design on and has submitted for review. The Prototype Release Level is for those R&D parts approved for prototyping and testing purposes. The Product WIP is the Product Work in Progress release level and consists of parts, assemblies and drawings currently under construction, or models being used for conceptualization, engineering changes, etc. for TACOM production programs. The Pre-Product Release Level contains models developed in accordance with this standard and submitted for review. The Product Release level contains models certified and approved in accordance with this standard and documented on an Engineering Release Record (ERR) by the Configuration Manager. The Legacy/Archive Release Level contains models previously approved but which are no longer needed in the Product Release or Prototype Release levels.

1.3 SECURITY: This system is intended for unclassified data only.

1.4 FOLDERS: The folder scheme used in INTRALINK is shown in figure 2. Further subfolders will be added at the request of the system configuration manager.

TACOM 3-D SOLID MODEL RELEASE SCHEME

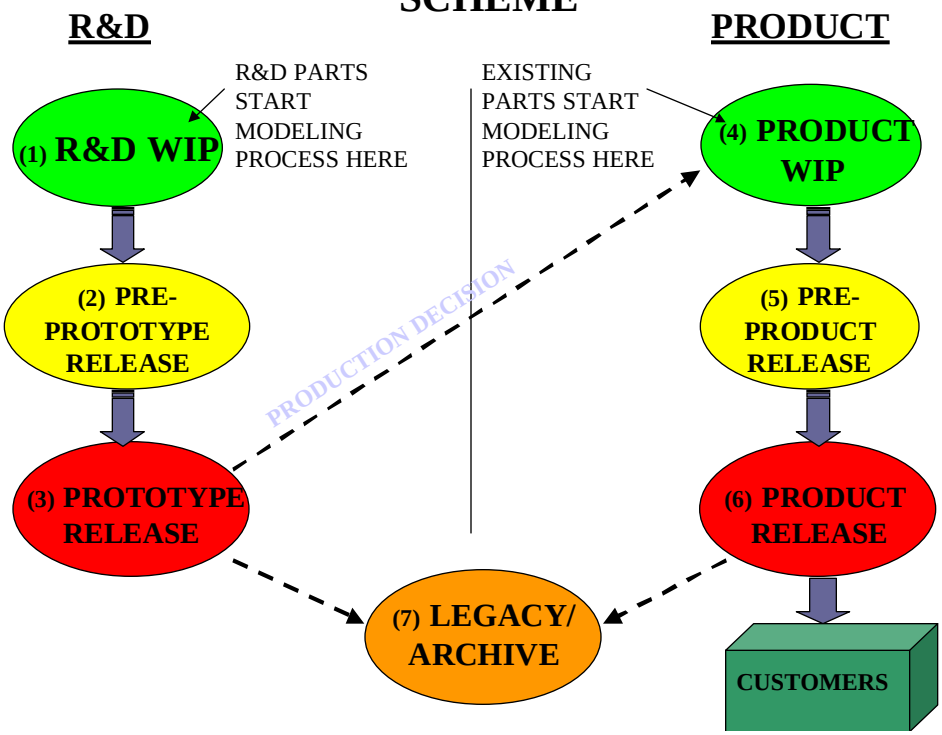

FIGURE 1. 3D/TDP Release Scheme

FOLDER SCHEME

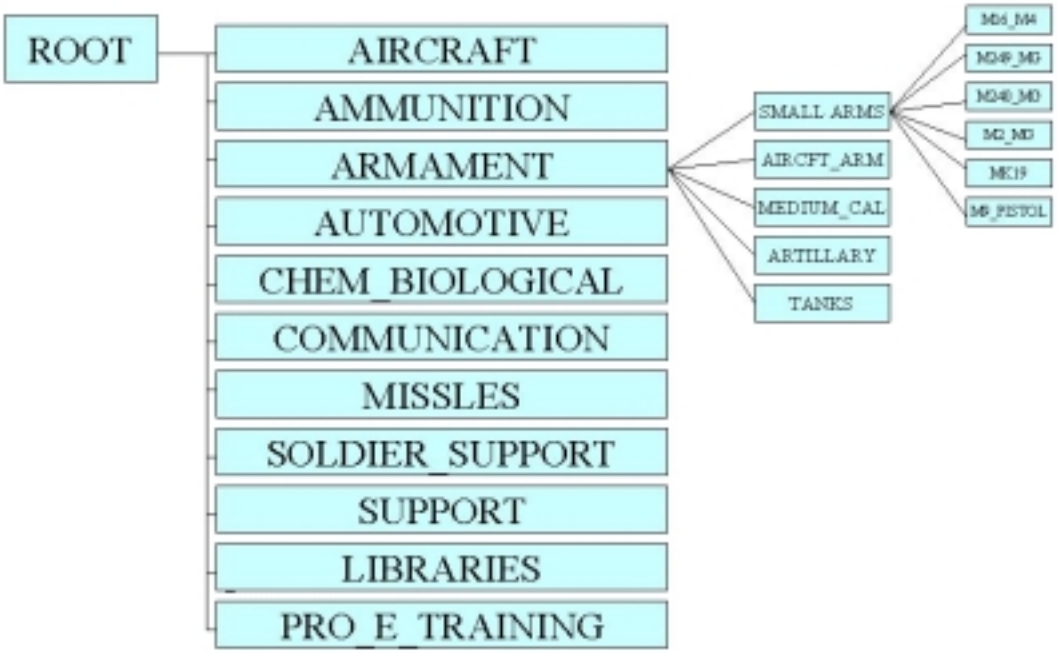

FIGURE 2. Folder Scheme

Chapter 2:
Configuration Management Representations

2.1 STANDARDS APPLICATION: The modeling requirements of this specification apply at the Product and Prototype release levels only. At the R&D and Production WIP release levels the modeling requirements are recommended but not required.

2.2 DRAWINGS REQUIRED: All parts and assemblies released into the Pre-Release and Product Release Levels shall have associated, fully defined 2D drawings (as required) also released into the database. When released into the Pre-Release and Product Release system, the models shall represent the latest design configuration (i.e., the latest revision drawing plus any outstanding ECPs).

2.3 ENGINEERING CHANGES:

2.3.1 Once in the Product Release system, models must be changed via approved Engineering Change Proposals (ECPs) only. The ECPs shall identify what files are affected such as drawing (.drw), part (.prt file) or assembly (.asm).

2.3.2 Each ECP shall result in a drawing and/or model revision, and conversely, each model or drawing revision shall be associated with a single ECP. In other words, a one to one relationship is maintained between revisions and ECPs.

2.4 CONFIGURATION MANAGEMENT: When a 3D/TDP associated 2D drawing is released into the database, that drawing becomes the master for configuration management purposes (i.e. drawings will not be input into the 3D/TDP and concurrently maintained in another system such as AutoCAD simultaneously.) Product drawings for JEDMICS shall then be generated and maintained from the 3D/TDP system.

2.5 REVISION SCHEME: The initial release of drawings, parts and assemblies will be the "-" revision level. Subsequent changes will be from a one to three digit alphabetic character (not including letters "I", "O" "Q", "S", "X" or "Z" (i.e., "A", "B", "C" … "AA", "AB", "AC", … "YYV", "YYW", "YYY".)

2.5.1 The part or assembly revision scheme is independent of its associated drawing and vice versa. For example, a part called "BRACKET", Part Number 12345, and its associated drawing are initially released at the "-" revision level. If a change is made to the model that does not affect the drawing, the model advances to the "A" revision level, but the drawing remains at the "-" level.

2.5.2 Multi-sheet drawings shall have a single revision number for all drawing sheets. The revision level shall be shown on drawing sheet 1 only. Both the drawing revision and model revision levels will be shown on the drawing sheet 1.

2.5.3 When legacy drawings are converted to solid model format, the solid model 2-D drawings shall be released at the next revision level. The solid model itself shall be release at the "-" revision level. For example, part Pin, Firing is at a "D" revision level when it is converted to solid model format. When released into the 3D/TDP system, the 2-D drawing developed from the solid model will be release at the "E" revision level (even if no design changes were made to the drawing) and the solid model which describes Pin, Firing will be released at the "-" level.

Chapter 3:
Part Modeling Requirements

3.1 PART MODELING PRACTICE

3.1.1 Parts must be functional and created in a logical sequence. Features should be simple and be created such that they are easily modifiable by another user at any time. Parts must have every feature that the real part is required to have. For example, plastic parts must contain every draft angle and radii detail expected on the real part. If text (e.g. part number) is stamped or molded into a part, that shall also be modeled. One exception is that threaded parts may be represented with cosmetic threads.

3.1.2 When possible, dimensions on models shall be from datums or surfaces, not edges or tangency points. When this is not possible, datum curve(s) or point(s) should be inserted to avoid parent/child related failures if the edge is broken or rounded. When creating a feature, sketch references should be chosen from the base feature or primary datums. This will help reduce unnecessary parent/child relationships.

3.1.3 Use of unnecessary datums should be kept to a minimum. Use datums "on the fly" when appropriate.

3.1.4 Parts should be able to stand on their own. While designing in assembly mode is often fast and easy to do, it creates parent/child relationships that are tedious to break. Features should be created only on the part to which they apply. An exception is features intended to be manufactured at assembly.

3.1.5 All models shall regenerate successfully with all features in the unsuppressed state. When parts are modified, the next higher assembly shall also be capable of being regenerated. Parts shall not contain buried features.

3.1.6 Rounds and fillets should generally be applied at the end of the model creation and placed on a separate layer.

3.2 TOLERENCES AND UNITS: All tolerances and unit systems shall be identified.

3.3 MATERIALS: Materials shall be identified on the part in two locations:

a. The material section (SETUP-MATERIALS) of Pro/Engineer. When possible, standard materials from the materials library shall be used. When a unique material is used, all material properties must be provided.

b. Under the materials note section of the 3-D notes which are applied to the drawing.

3.4 SURFACE FINISHES: Surface finishes shall be shown on the part as required.

3.5 3D NOTES: Sufficient 3D notes shall be used to describe the part and part features. If notes are intended to apply to specific features they shall be attached to those features. The drawing Distribution Statement shall be included in a 3D note called "Distribution_Note". If a proprietary legend is required on the drawing, this legend shall be entered on the model as a 3D note entitled "Proprietary_Note".

3.6 NOTE_0: NOTE_0 shall be used to document design intent. Design intent is any optional information the designer thinks is valuable to those who might view the model at some future point. An example of a design intent note is as follows: "THIS PART IS MADE OF ALUMINUM 7075-T6. CARBON STEEL WAS TESTED BUT CORRODED DURING USE. USE OF STAINLESS WAS CONSIDERED BUT NOT PURSUED DUE TO TIME CONSTRAINTS."

3.7 DEFAULT DATUMS: All parts will start with default datum planes and a default coordinate system. The default datum planes shall be named "FRONT", "TOP", and "SIDE". All parts should have a datum

coordinate system set with x, y and z correctly pointing to the part's final orientation (i.e., "FRONT" in the x-y plane, "SIDE" in the y-z plane, and "TOP" in the x-z plane). These default datum planes and coordinate systems are generated automatically when the TACOM start part is used for part creation.

3.8 FILE NAMES: Parts shall be named using the following convention:

CAGE_PartNumber.prt i.e. 19200_8448510.prt

3.9 MODEL VIEWS: All models shall have separately named views defining the "TOP_VIEW", "BOTTOM_VIEW", "LEFT_VIEW", "RIGHT_VIEW", "BACK_VIEW", "FRONT_VIEW", and "DEFAULT" (isometric) views. These views are generated automatically when the TACOM start part is used for part creation. Parts should be created such that the named views correspond to the actual part being created.

3.10 SHEETMETAL PARTS: Sheetmetal parts should be created in Pro/SHEETMETAL or be capable of being imported into Pro/SHEETMETAL. Sheetmetal parts should be started as solid parts using the TACOM_START_PART and the standard naming convention, then converted to sheetmetal by starting the SHEETMETAL application.

3.11 DATUM NAMES: Datum planes, other than default, should be named using appropriate and logical names. This allows SEL BY MENU functionality when referencing any datums even when datums are layered off or not displayed.

3.12 RELATIONS: All relations shall have a comment describing the relation. Relations shall have no errors upon regeneration.

3.13 DEFORMED PARTS: Parts that are deformed at assembly must be created using the Family Table command. The Family Table must contain all variances of deformation to include an unassembled "free state". As an example, a helical spring that has a 1.0-inch free length and is assembled to a 0.8 inch compressed length shall have two instances in a family table. The first instance is the "generic" or "free state" of 1.0 inch shown on the drawing, the second instance is of the "assembled state" of 0.8 inches shown on the next higher assembly.

3.13.1 FAMILY TABLE NAMING: Family table instances must be uniquely named and in the case of Deformed Parts they should indicate where they are used. The "Generic" family table instance should be used on the drawing. A deformed instance that is to be used on a higher-level assembly should be named "CAGECODE_PARTNUMBER_IN_PARTNUMBER" where the second "PARTNUMBER" is the higher-level assembly.

Example: 19200_12345678_IN_87654321

3.14 FORGINGS AND CASTINGS:

3.14.1 Parts made from forgings or castings, in which the as forged/cast and machining dimensions are depicted using a single part number shall have the forging/casting completely modeled and separately layered from the machining dimensions, i.e. the as forged/cast part shall be completed in the model tree before the machining features are added. The machining features shall then be separately layered on a layer named MACHINED_FEATURES, so that the machining features can be suppressed and the as forged/cast part viewed.

3.14.2 Parts made from forgings or castings in which the forged/cast part is shown on a separate part number must be created with the "forged/cast" part and the "machined" part as separate parts. The "machined" part is made from the "forged/cast" part using the Merge command in the assembly mode or another suitable method as long as the machined part is parametrically linked to the cast part. See appendix C for the Merge procedure.

3.15 LAYERS: The following layers shall be used in part mode (these are automatic when the TACOM start part is used). Other layers may be added as needed, for example, a datum required on a drawing may be placed on a separate layer so that it can stay on while the other datums are layered off.

	Layer name	Layer Features	Default Suppress State
a.	DATUMS	Datum planes, points, curves, coordinate system	ON
b.	HOLES	All holes.	ON
c.	RNDS_CHAMFERS	Rounds and Chamfers	ON
d.	DIMENSIONS	All Dimensions	ON
e.	NOTES	All part notes	ON
f.	DRAFTS	Drafts	ON
g.	SURF_FINISHES	Surface Textures and finishes	ON
h.	THREADS	Cosmetic and actual threads	ON
i.	SURFACES	All Surface features and Quilts	ON

3.16 PARAMETERS: The following parameters are required when entering a part or assembly into the 3D/TDP Product Release Level (Using the "SETUP-PARAMETERS" Command):

	Parameter Name	Parameter Description	Format Example	Input
01	PART_NUMBER	Item part number	**12556974**	REQ
02	CAGE_CODE	Original Design Activity CAGE code	**19204**	REQ
03	CURRENT_CAGE_CODE	Current Design Activity CAGE code	**19200**	-
04	NOMENCLATURE	Part Name	**RIFLE, BOLT, CARRIER, ASSY**	REQ
05	DRAWING_DATE	Date Pro/E drawing created	**1998-12-24**	REQ
06	DRAWN_BY	Person who created Pro/E drawing	**F. FLINSTONE**	REQ
07	ORG_DRAWING_DATE	Date original drawing created	**1969-07-28**	-
08	MODELER_NAME_1	Primary Pro/E modeler	**Y. SAM**	REQ
09	MODELER_NAME_2	Secondary Pro/E modeler	**L. FOGHORN**	-
10	MODELER_NAME_3	Auxiliary Pro/E modeler	**B. BUNNY**	-
11	CHECKER_NAME	Person that checked original	**G. JETSON**	REQ
12	ENGINEER_NAME_1	Primary design engineer	**Y. BEAR**	REQ
13	ENGINEER_NAME_2	Secondary design engineer	**R. SMITH**	-
14	QA_ENGINEER_NAME	Quality Assurance engineer	**D. DWAG**	-
15	MATERIAL_ENGINEER	Person who OK'd material for design	**W. COYOTE**	-
16	DRAWING_APPROVAL_NAME	Person who approved drawing	**H. HONCHO**	REQ
17	DRAWING_APPROVAL_DATE	Date drawing approved	**1999-12-25**	REQ
18	DESIGN_APPROVAL_NAME	Person who approved design	**E. CARTMAN**	-
19	DESIGN_APPROVAL_DATE	Date design approved	**2000-10-25**	-
20	ERR_ECP_NUMBER	ECP that caused current revision	**G9S3001**	REQ
21	ERR_DATE	Date current revision released	**1998-10-21**	REQ
22	ERR_ECP_APPROVAL	Approving official of latest ECP		
23	MODEL_REV	Revision of current model	**A**	REQ
24	DRAWING_REV	Revision of current drawing	**C**	REQ
25	CONTRACT_NUMBER	Contract #-original design contract	**DAAE20-99-C-0001**	-
26	DESIGN_CONTRACTOR	Original design contractor	**DESIGN INC.**	-
27	MANUFACTURER_NAME	Current/last manufacturer	**ACME**	-
28	USED_ON_1	First used on item	**M4**	REQ
29	USED_ON_2	Second used on item	**M4A1**	-
30	USED_ON_3	Third used on item	**M16**	-
31	USED_ON_4	Fourth used on item	**M16A1**	-
32	USED_ON_5	Fifth used on item	**M16A2**	-
33	USED_ON_6	Sixth used on item	**M16A3**	-
34	USED_ON_7	Seventh used on item	**M16A4**	-
35	USED_ON_8	Eighth used on item		-
36	NEXT_ASSY_1	First next higher assy part no. (must match USED_ON_1 above	**1345678**	REQ

37	NEXT_ASSY_2	Second next higher assy part no.	**2345678**	-
38	NEXT_ASSY_3	Third next higher assy part no.	**3445678**	-
39	NEXT_ASSY_4	Fourth next higher assy part no.	**4345678**	-
40	NEXT_ASSY_5	Fifth next higher assy part no.	**5545678**	-
41	NEXT_ASSY_6	Sixth next higher assy part no.	**6545678**	-
42	NEXT_ASSY_7	Seventh next higher assy part no.	**7645678**	-
43	NEXT_ASSY_8	Eighth next higher assy part no.	**8945678**	-
44	NSN	National Stock Number (if assigned, type "NONE" otherwise)	**1005-01-123-4567**	-
45	CATALOG_NOMENCLATURE	Nomenclature assigned by supply	**SPRING, HELICAL**	REQ
46	DISTRIBUTION_CODE	Drawing Distribution Statement Code	**A**	REQ
47	PROPRIETARY_CODE	Code id for proprietary category	**M16_RIFLE_PROPI**	**
48	REMARKS	General Remarks		
49	OZONE_DEPLETING_CHEMICAL	YES-NO PARAMETER, Ozone depleting chemical	**NO**	
50	SPECIALTY_METAL	YES-NO PARAMETER, Specialty Metal	**NO**	
51	HEAVY_PHOSPHATE	YES-NO PARAMETER, Contains Phosphate	**NO**	
52	JEWEL_BEARING	YES-NO PARAMETER, Contains Jewel Bear	**NO**	
53	SOURCE_CONTROL	YES-NO PARAMETER, Source Control	**NO**	
54	CASTING	YES-NO PARAMETER, Casting/Forging	**NO**	
55	PRO_E_REVISION	Pro/Engineer Revision number	**REVISON 2000i2**	REQ
56	UNIT_WEIGHT	Weight_of_Item (from Relations)	**4.247**	
57	ECP_DATE	Date ECP Approved	**1998-10-21**	REQ

3.16.1 The following parameters are also required and are automatically generated when the TACOM start part and assemblies are used:

	Parameter Name	Parameter Description	Format Example	Input
58	AGENCY	Agency or service (from Relations)	US ARMY	
59	DESIGN_ACTIVITY	Design Activity (from Relations)	Armament Research, Development and Engineering Center	
60	DESIGN_ACTIVITY_LOCATION	Design Activity City, State and Zip Code (from Relations)	Picatinny Arsenal, Dover NJ 07806	
61*	NOMENCLATURE_1	First NB_1 characters of Nomenclature (for drawing) (from Relations)		
62*	NOMENCLATURE_2	Second NB_2 characters of Nomenclature (for drawing) (from Relations)		
63*	NOMENCLATURE_3	Remaining characters of Nomenclature (for drawing) (from Relations)		
64*	NB_1	First line break in nomenclature (from Relations)	20	
65*	NB_2	Second line break in nomenclature (from Relations)	40	
66*	NOMENCLATURE_LENGTH	Number of characters in nomenclature	60	

* Non-Designated Parameters

Other parameters, layers and datums may be added as needed during modeling.

3.16.2 The parameter “CAGE_CODE” refers to the original design activity. If the current design activity is different than the original design activity then the current one is entered into the “CURRENT_CAGE_CODE” field. A table must also be placed on the drawing showing the current design activity cage code (see sample below).

Chapter 4
Assembly Creation Requirements

4.1 ASSEMBLY MODELING PRACTICE

4.1.1 Assemblies shall have all parts of the assemblies, including pins, screws, nuts, washers, springs etc. defined. This includes items such as decals, warning labels, etc., which are required to be attached to the part.

4.1.2 Assembly sequence should follow the intended manufacturing assembly sequence. The mate, offset, align method shall be used when ever possible.

4.1.3 Assembly features should not be used for such things as part features. For example, a cut, slot or hole assembly feature should not be used on two or more parts to create mounting holes (unless these holes are intended to be created at assembly). These features do not show up while in part mode.

4.1.4 Parts in an assembly should be assigned a unique color for ease of viewing the assembly.

4.2 ASSEMBLY TOLERENCES/UNITS: All tolerances and unit systems shall be identified.

4.3 3-D NOTES: Sufficient 3D notes shall be used to describe the assembly and assembly features. If notes are intended to apply to specific features, they shall be attached to those features.

4.4 ASSEMBLY DEFAULT DATUMS: All assemblies will start with default datum planes and a default coordinate system. The default datum planes shall be named "FRONT_ASSY", "TOP_ASSY", and "SIDE_ ASSY". All assemblies should have a datum coordinate system set with x, y and z correctly pointing to the assemblies' final orientation (i.e., "FRONT_ASSY" in the x-y plane, "SIDE_ASSY" in the y-z plane, and "TOP_ASSY" in the x-z plane). These default datum planes and coordinate systems are generated automatically when the TACOM start assembly is used for part creation.

4.5 FILE NAMES: Assemblies shall be named using the following convention:

CAGE_PartNumber.asm i.e. 19200_8448511.asm

4.6 MODEL VIEWS: All models shall have separately named views defining the "TOP_VIEW", "BOTTOM_VIEW", "LEFT_VIEW", "RIGHT_VIEW", "BACK_VIEW", "FRONT_VIEW", and "DEFAULT" (isometric) views. These views are generated automatically when the TACOM start assembly is used for assembly creation. Assemblies should be created such that the named views correspond to the actual assembly being created.

4.7 ASSEMBLY LAYERS: The following layers shall be used in assembly mode (these are automatic when the TACOM-ARDEC start assembly is used):

	LAYER NAME	LAYER FEATURES	DEFAULT SUPPRESION STATE
a.	ASSY_DATUMS	Assembly datum planes, points, curves and coordinate systems	ON
b.	ASSY_HOLES	Assembly holes	ON
c.	ASSY_NOTES	Assembly notes	ON
d.	ASSY_DIMENSIONS	Assy dimensions	ON

4.8 BULK ITEMS: Bulk items (i.e., adhesives, lubricants, etc) shall be identified on assemblies in order that they appear in bills of material.

Chapter 5
Drawing Creation Requirements

5.1 DIMENSIONS

5.1.1 Drawings shall display dimensions that are directly parametric to the model upon which they are based. Specifically, these values must be driving or driven dimensions. In no case, should dimension values be entered as text notes which are not parametric to the base model.

5.1.2 Geometric tolerances shall be attached to the base dimensions when applicable and not added as unattached notes

5.2 ISOMETRIC VIEWS: When possible, 3D/TDP associated 2D drawings should have an isometric view for reference purposes.

5.3 FORMATS: Standard 3D/TDP drawing formats shall be used and are available from the formats library. Standard drawing setup files (.dtl) files are also available for each format and shall be used.

5.4 STANDARDS: All drawings shall be fully defined in accordance with ASME Y14.5, ASME Y14.1 and MIL-STD-100. To assist in this requirement, standard drawing setup files have been created for each drawing format. These are available in the INTRALINK Standards directory.

5.5 NOTES: All notes on drawings must originate as 3D notes in .prt and .asm files. The numbered notes on the drawings shall be identified in the .prt and .asm files as "NOTE_1", "NOTE_2", etc. When applicable, notes should be attached to the dimensions or features to which they apply.

5.6 DRAWING VIEWS: Draft geometry shall be attached to the drawing view to which it belongs. (This is accomplished using the **TOOLS > DRAFT VIEW > SET CURRENT** command.) Views should be capable of being moved using the **VIEW > MOVE VIEW** command with no entities of the view being left behind.

5.7 DRAWING LINE-WEIGHTS: The following line weights and colors shall be used:

White, Normal Weight: Object lines
Yellow, Thin Weight: Leader lines
Gray and/or dashed: Hidden Object lines
RED, Bold: Special lines needing extra emphasis.

5.8 QUALITY ASSURANCE INFORMATION: The preferred method for identifying Quality Assurance Information is on the product drawing instead of having a separate QAP sheet. The quality.sym included in the 3D TDP files is provided for this purpose. The definition of this symbol is shown on drawing # 19200_12993884.

5.9 DISTRIBUTION STATEMENT: The drawing distribution statement must appear on each sheet of a multi-sheet drawing. The distribution statement is in the form of a note and may be retrieved with the mapkey “RET_NOTES”.

5.10 REVISION BLOCK: The revision block contains the parameters MODEL_REV, DRAWING_REV, ERR_ECP_NUMBER, ERR_ECP_DATE, and ERR_ECP_APPROVAL on the last line of the table. These parameters source from the model. Previous revisions should be entered into the table manually. The table may be expanded as necessary by using **TABLE > MOD ROWS/COL > INSERT** in the drawing mode.

Chapter 6
Database Management

6.0 PRODUCT DATA MANAGEMENT SYSTEM: The Product Data Management system used to maintain the 3-D TDP system will be Pro/INTRALINK. The metadata for the system will be maintained at Picatinny Arsenal, Dover NJ with replicated sites at Rock Island Arsenal and TACOM-Warren. The sites shall be configured as shown in figure 3.

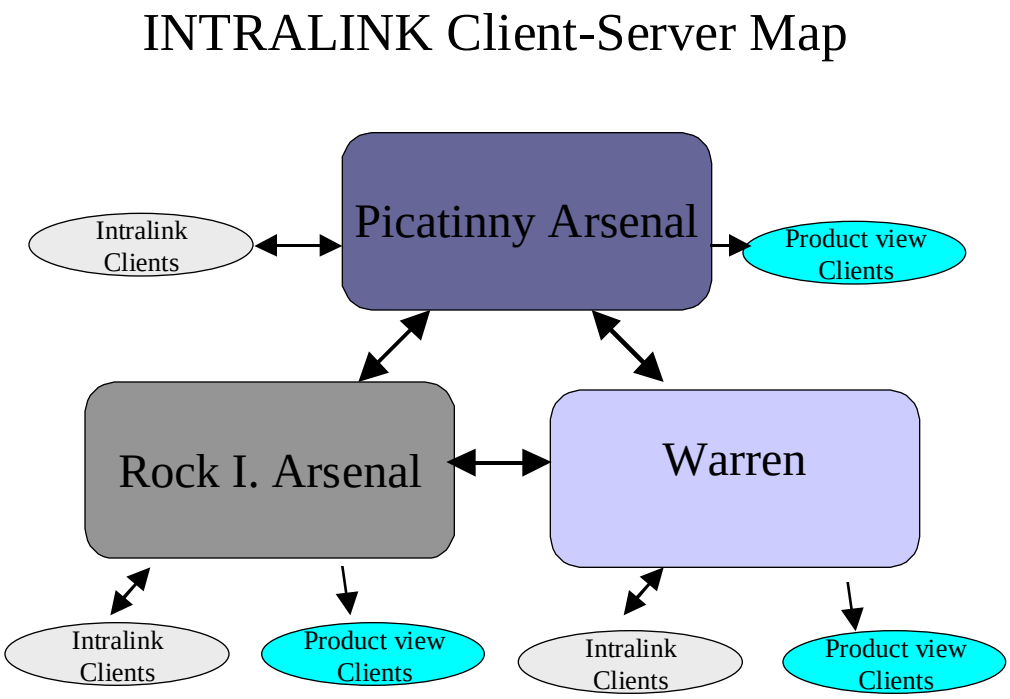

Figure 3. INTRALINK Client-Server Map

6.1 User Authorizations: User authorizations shall be set for each folder protect proprietary data and prevent unauthorized modifications.

Chapter 7
Libraries

6.1 MATERIALS LIBRARY: Materials shall be assigned from the materials library. If a needed material is not in the materials library, it shall be added to the library before use.

6.2 TEXTURES LIBRARY: Contains standard textures.

6.3 FASTENERS LIBRARY: Standard hardware such as pins, screws, nuts, etc. shall originate from the standard parts library.

6.4 CONNECTORS LIBRARY: Contains all electrical connectors.

6.5 COMMERCIAL ITEMS LIBRARY: Contains models of commercially available models. For example, a bearing company provides Pro/Engineer models of their items which can be incorporated directly into an ARDEC assembly.

6.6 TOOLING LIBRARY: Contains models of tooling and fixtures.

6.7 MOCKUP COMPONENTS LIBRARY: Mockup components are parts that are modeled only as placeholders in a design. An example would be a radio which will be installed into a vehicle where the radio exterior features need to be shown for space/weight requirements only and it would be uneconomical or unnecessary to model every external and internal feature of the radio. These mockup components shall be modeled when necessary and located in the Mockup Component library and shall be reasonably accurate in exterior features, weight and center of gravity.

6.8 FORMATS LIBRARY: Drawing formats shall be placed in the formats library. ARDEC standard formats shall be named A_TACOM_MET_1_0.frm, B_TACOM_ENG_1_0.frm, etc.

6.9 STANDARDS LIBRARY: Contains 3D/TDP standards document, standard start part and start assemblies and standard drawing setup files (A_TACOM_ENG_1_0.dtl, etc.).

7.0 NOTES: Proposed changes to this document should be submitted to Commander, ARDEC, AMSTA-AR-ES, ATTN: Mr. Jeff Windham or Mr. Dave Collum, Rock Island, IL 61299-7300.

Appendix A
Glossary of Pro/ENGINEER & INTRALINK terminology

Pro/ENGINEER Terminology

- **Coordinate system** - The intersection of the three datums is the origin of the part's coordinate system; coordinate axes are formed by the intersections of each datum pair. Users select axes convenient to the creation of their part for alignment of some of the part's features. Your view is normal to the sketch plane (unless you want to sketch in 3-D – not recommended for the beginner). Protrusions are created normal to the sketch plane, toward you.
- **Datum** - A datum is a reference plane, used to define where in space the part is to be created. The space is completely defined by three orthogonal datums: users select two, and Pro/E generates the third. One datum is then selected as a sketch plane, on which the outline is drawn.
- **Feature-based** – you design a part using *features* (e.g., extrusions, holes, slots, rounds, etc.) instead of the part's geometric characteristics (i.e., individual arcs and lines).
- **Fully associative** - if a given *parameter* is changed on a certain *feature*, any *associated* dimension(s) are automatically changed accordingly.
- **Parametric** – the shape of the created part is based on the dimensions of its features (or, *parameters*), and those dimensions can be related to (or, *associated with*) one another (e.g. edges can be parallel to or normal to one another; lengths or angles can be equal to one another).

Pro/INTRALINK Terminology

Pro/INTRALINK information is organized by objects. Each object is uniquely identified within Pro/INTRALINK by the following four attributes:

- **Name**-A name is assigned to all versions of a standard object, for example, piston.prt or crankshaft.asm.

- **Branch**-The branch defines multiple alternatives for a standard object. Each standard object automatically belongs to a default branch, called main. You can optionally create new branches, descending from the main branch, for each object.

- **Revision**-The revision is a company-wide identifier defined by the Pro/INTRALINK Administrator to indicate an object's level of maturity.

- **Version**-The version is a number that is used to distinguish between multiple iterations of an object. The version number starts at zero and ascends by ones. When the revision of an object is increased, the version number is reset to zero.

  The full identifier for a standard object, therefore, can be written as follows:

name	branch	revision	version
a33956.prt	main	1	1

  If you create a new branch for an object or increase the object revision or version, you are in fact creating a new object version. For example, if you increase the revision of the preceding object, you create the following object:

name	branch	revision	version
a33956.prt	main	2	0

As these are two distinct object versions, they enable you to keep a history of object changes. Therefore, if you do not like the changes you made to a later version of an object, you can retrieve the earlier version.

- **Release Levels** - A standard object has an additional attribute, called its release level. The release level indicates at which stage of a lifecycle an object version exists. Release levels are defined by the Pro/INTRALINK Administrator.

- **Baselines** - Since it is common to work on many objects simultaneously as a unit, it is helpful to have a quick way to identify a grouping of objects. That is the role of baselines. Just as you can identify an object by its four identifiers, you can identify a grouping of objects by a baseline. A baseline can be viewed as a named snapshot of grouped objects at a particular point in time.

Appendix B
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Appendix C
Forging/Casting Procedures

Parts made from forgings or castings in which the forged/cast part is shown on a separate part number must be created with the “forged/cast” part and the “machined” part as separate parts. The “machined” part is made from the “forged/cast” part using the Merge command in the assembly mode or another suitable method as long as the machined part is parametrically linked to the cast part

Merge Method

1. First, create the forged/cast part in the "**PART**" mode of Pro/E. Next, create a new part for the machined portion. This new part will only contain the default datum planes. Be sure to save the machined part, with just the default datum planes, and exit out of the **PART** mode.

2. Create a new assembly for the forged/machined part. Next go to "**Component** > **Assemble**" and pull in the forged/cast part. Then again go to "**Component** > **Assemble**", and pull in the machined part (this will only contain three default data planes). Align the default datum planes of the machined part to the three planes of the forged/cast part.

3. Next go to "**Component** > **Adv Utils** > **Merge > Select by Menu**", and pick the machined part, "**Done Sel**". Then pick the forged/cast part from the "**Select by Menu**" and "**Done Sel**".

4. Keep the defaults "**Reference**" and "**No Datums**". Reference will update parts if future changes are made. Copy will not update the parts. No Datums the part will merge to base part without datums.

5. The parts have now been merged together. You can now "**Save**" and get out of "**Assemble**" mode and return to "**Part**" Mode. If you now pull up the machined part, you will see the forged/cast part appear. You can now make machining changes to the forged/cast parts. The original "forged/cast" part will not be changed, however if changes are made to the "forged/cast" part they will automatically update the machined part.

Appendix D
Start Part and Start Assembly Procedures

1. First, create a new part using the "**Create New Object**" icon or "**FILE-NEW**".

2. Name the new part in accordance with the standards, i.e., CAGE_PartNumber.

3. Click on the "**Copy From**" icon and chose the TACOM_START_PART_1_0.PRT or TACOM_START_ASSY_1_0.ASM as appropriate.

4. Click on the "**START_MODEL**" icon and fill in parameters and appropriate unit system. If a parameter is unknown, skip it and fill in later using the "**SETUP > PARAMETERS > MODIFY**" command.

5. Start modeling.

6. For sheetmetal parts, follow the procedures above for solid parts, then convert the solid part to sheetmetal by entering the Pro/SHEETMETAL application.

Appendix E
Checking Procedures for 3D Technical Data Packages

Overview

Before a 3-D TDP can be certified each part, drawing and assembly must be checked for accuracy and completeness. All 3-D parts must be checked to insure accuracy of notes, parameters, datum's, dimensions and tolerances, geometric tolerances, surface finishes, and relations. In addition, parts must be checked for completeness to insure all features have been defined and they have been appropriately aligned. The parts, drawings and assemblies shall be checked for compliance with the TACOM 3D Technical Data Package Standards.

Drawings must be checked to insure all views have been defined, all title block information, revision block information, notes, dimensions, tolerances, surface finishes, and geometric tolerances have been imported from the part.

Assemblies must be checked to insure all parts have been imported and assembled correctly, and each part must be colored separately to visually denote the part.

If any discrepancy is discovered with the part, drawing or assembly, the discrepancy is to be documented and provided to the modeler to be corrected.

Parts

Before the part is to be checked, use the **Info > Regen Info** to ensure the part has been built using a layered sequence. The following items need to be checked for each part:

a. Parameters – Parameters can be found under the **Setup** command. Insure that each required parameter has been entered, and all parameters have been entered correctly.

b. Notes – Notes can be found under the **Setup** command. Insure all notes have been entered and no typographical errors exist. Insure all drawing notes are pointing to the correct surface or edge.

c. Datum Planes – All datum planes should be layered off when submitted for checking. To insure all datums have been defined turn the datum's on in the **Layer** command.

d. Dimensions, Tolerances and Geometric Tolerances – Before tolerances can be displayed go to the **Environment** command and check the **Tolerances** block and **Apply**. To check all dimensions and tolerances go to **Modify**, **Dimensions** and individually select each feature from the model tree. Insure that all dimensions, tolerances and geometric tolerances have been accurately defined.

e. Surface Finishes – Surface finish can be found under the **Setup** command. Insure all surface finishes have been defined and no typographical errors exist.

f. Datum Curves – If datum curves exist, they should be layered off when submitted for checking. Datum curves are used to define datum target areas, areas for hardness checks, and knurling. Insure all datum curves have been accurately defined.

g. Cosmetic Threads – All threads have been created as cosmetic features. Insure all threads have been assigned a cosmetic feature and they are accurately defined.

h. Family Tables – Springs contain family tables to represent the part in the compressed and the free state. Insure that the family table contains the correct dimensions needed to accurately define the part in both instances.

Drawings

The following items need to be checked for each drawing:

a. Title Block – Insure all parameters have been imported from the part.

b. Revision Block – Insure the revision block has been updated correctly. The revision block shall contain the latest revision level. Once the ECP has been approved to release the new 3-D TDP the revision block will be updated to reflect the new revision.

c. Notes – Insure all general notes, including the distribution statement, have been placed on the drawing. Insure drawing notes are correct and located in the correct position.

d. Views – Insure that all views needed to accurately define the part have been placed on the drawing. Some views may differ from the current drawing views. (i.e. views may be placed in a slightly different location for clarity, cross-section views may have cross section lines running in different directions, etc.)

e. Dimensions, Tolerances and Geometric Tolerances – Insure all dimensions to define the part have been imported into the drawing. These dimensions shall be parametric. Some dimensions may be placed in different locations on the view to aid clarity. In addition, insure that all dimensions are driven or driving dimensions.

f. Leader Lines – Insure leader lines do not run into the part, but are offset for clarity.

g. Surface Finish – Insure all surface finish symbols have been applied.

Assemblies

The following items need to be checked for each assembly:

a. Importing Parts – Insure all parts are present and fully constrained.

b. Assembling – Insure all parts have been assembled correctly in the correct sequence.

c. Coloring – Insure each part has been colored separately for clarity. It is acceptable to color similar items for easy identification. (i.e. coloring all springs the same color, etc.)

Model Check List

- All known information is entered into the parameters using START_MODEL macro.
- Material is assigned to part.
- Finish note is completed.
- Hardness note is completed.
- All dimensions made with correct accuracy and tolerances.
- All required geometric tolerances and datum references made.
- All required surface finishes added.
- All required sections made for the drawing.
- Proprietary legend note made if required.
- Datum planes, other than default, named using appropriate and logical names. (Ref: paragraph 3.11 of Modeling Standards)
- Parts that are deformed at assembly created using Family Table command. (Ref: paragraph 3.13 of Modeling Standards)
- The part_name.bmp file is a correctly sized picture of the part with notes layered off and tolerance display turned off.

Drawing Check List

- Correct format is used.
- Isometric view is included.
- Tolerance values included in title block.
- All dimensions are directly parametric to the model. (Ref: Paragraph 5.1.1 of Modeling Standards)
- Proprietary note is included on drawing if necessary.
- Distribution statement is included.
- Each view has correct amount of information shown (axes, hidden lines, tangent lines, etc.)
- All views are scaled correctly.

Assemblies

- All parts of the assembly are fully constrained.
- Assembly sequence of the model follows the assembly sequence of actual parts.
- Parts are separately colored for ease of viewing.

Appendix F
Procedures to update parts to latest TACOM standards

These procedures can be used to update most parts, drawings and assemblies to the latest TACOM standards but they should be checked carefully.

1. Open part and drawing (if applicable) and top level assembly(s) containing that part all at once. Rename to proper naming convention (CAGE_PARTNUMBER). Save part, drawing and top level assembly(s). Close drawing and assembly(s).

2. Delete any parameters not listed in the TACOM 3D Standards

3. Run mapkey **APPLY_PARAMETERS** and hit <RETURN> all the way thru parameter list.

4. Run mapkey **RELATIONS**.

5. Hit **START_MODEL** mapkey and fill in all known values.

6. Make sure default datum plates are named FRONT, TOP and SIDE. If not, run mapkey **RENAME_DATUMS**.

7. If default views are not defined run mapkey **CREATE_VIEWS**.

8. Run Mapkey **CREATE_NOTES**.

9. Save Part.

10. Open Drawing, remove old format, apply new TACOM standard format (**SHEETS > FORMAT > ADD/REPLACE**). Apply TACOM drawing setup values (**ADVANCED > DRAW SETUP > RETRIEVE**). Click on **RET_NOTES** mapkey and click twice in body of drawing.